ELSEVIER

Available online at www.sciencedirect.com

ScienceDirect

Procedia Technology 24 (2016) 347 – 354

Procedia Technology

International Conference on Emerging Trends in Engineering, Science and Technology (ICETEST - 2015)

Modelling of Energy Dissipation During Transient Gaseous Cavitation

Resmy K[a], K S Sumam[b*], N Sajikumar[b]

[a]*Student, Govt. Engineering College, Trichur, 680009, India*
[b*]*Professor, Govt. Engineering College, Trichur, 680009, India*
[b]*Professor, Govt. Engineering College, Trichur, 680009, India*

Abstract

Hydraulic transient/ waterhammer analysis is important in the operation stage of an existing piping system for the diagnosis of malfunction problems or the causes of pipe bursts. Classical waterhammer equations cannot represent the energy dissipation phenomena after the waterhammer peak. Therefore, it is extremely important to use accurate hydraulic transient models which can incorporate additional dissipative effects in the analysis. In this study, effect of gaseous cavitation is considered for the modelling of transient flow during valve closure using a 2D approach in cylindrical coordinates. This developed model could predict the first water hammer drop and gas release. However extra damping is observed in the subsequent peaks compared to the experimental results which necessitates further investigation.

Peer-review under responsibility of the organizing committee of ICETEST – 2015

Keywords: *Transient flow; Gaseous cavitation; Two dimensional model.*

1. Introduction

Transient flow is the intermediate stage flow between two steady state flow conditions. It generally occurs whenever the flow changes abruptly with time. The occurrence of transient flow induces large pressure forces and rapid fluid accelerations into a water distribution system. When the velocity of flow changes rapidly due to the change in operating condition of the flow controlling components, like closure of a valve, pump start up/stop etc, causes a pressure wave which travels throughout the system. The pressure wave starts to travel from the point of generation towards the other end and gets deflected back (in a closed system). This to and fro motion of the pressure

* Corresponding author. Tel.: +919496167977.
E-mail address: sumam@gectcr.ac.in

Peer-review under responsibility of the organizing committee of ICETEST – 2015
doi:10.1016/j.protcy.2016.05.046

wave continues for a long time. During the motion of the pressure wave, the performance of the whole system components gets disturbed which manifests the malfunctioning of the hydraulic equipments in a pipe network.

During transient flow when the pressure in the pipe becomes lower than the saturation pressure of flowing liquid, the dissolved gases get released which will flow along with the already present free gas. When the pressure in the flowing system drops, two mechanisms will occur. The first stage is gaseous cavitation and second is vaporous cavitation. Gaseous cavitation occurs when the pressure drops below the saturation pressure but above the vapour pressure. But, when the pressure drops to the vapour pressure of the liquid, vaporous cavitation occurs. i.e. conversion of liquid phase to vaporous phase. Because of these effects, there will be free gases in the flowing system which will reduce the transient wave speed. Sometimes the transient flow event can be extremely destructive if the magnitude and velocity of the pressure wave exceeds the capacity of system in which it takes place. This necessitates the prediction of the transient wave pressures. But the solution of recurring transient flow problems is not easy, and is generally only achievable with sophisticated software simulations.

Generally the computation for transient analysis is done on the basis of classical water hammer equations which cannot represent energy dissipation in an effective manner. i.e. discrepancy is observed between the actual and computed pressures especially when time progresses. This difference between the observed and calculated pressure may be due to energy dissipation. Therefore, in order to represent the energy dissipation during transient flow additional dissipative effects are to be incorporated.

[1] illustrates about a one-dimensional mathematical model which explains the behaviour of gas–liquid mixture transient flow. Hence, the purpose of their study is to investigate numerically the nonlinear behaviour of the transient homogeneous two-phase flow in pipes.

[2] presented a methodology which accounts non-friction energy dissipation in transient cavitating flows. The effect of gaseous cavitation on thermic exchange between gas bubbles and the surrounding liquids is described with the help of a 2D model.

[3] gives an alternate approach for modelling transient vaporous cavitation by considering the variable fluid property concept. In this study, the simulation of cavitating flow was carried out by using the continuity and momentum equation for the water vapour mixture, transport equation for the vapour phase.

[4] introduced a new discrete vapour cavity model (DVCM) to evaluate the column separation phenomena in hydraulic transients. In this study, they assumed that the calculated cavity volume in several computational pipe cross sections moves to one main cross section.

[5] discuss about the 2D model for analyzing transient cavitating pipe flow. The model considers the conservation form of continuity equation which allows simple numerical solution. 1D and 2D models are used to quantify the effect of friction in the simulation of experimental data. But the 1D model failed to reproduce the experimental results after the first peak. These studies do not consider the dissipative capacity of gas release.

Hence, the aim of this paper is to quantify the gas release as an additional dissipative effect and examine the energy dissipation in transient gaseous cavitating flow.

2. Mathematical Model

For gaseous cavitation modelling, the liquid is considered as a homogeneous two-phase air water mixture and the analysis is based on the following assumptions [2, 3]

1. Gas bubbles are distributed throughout the pipe and they are very small compared to pipe diameter;
2. Difference in pressure due to surface tension across a bubble surface can be neglected;
3. Momentum exchange between gas bubbles and surrounding liquid is negligible, so that gas bubbles and liquid have the same velocity.

2.1. Governing Equations

An important feature for gaseous cavitating flow modelling is that, it considers the flowing liquid as a homogeneous mixture of liquid and gas. For this study, modelling is carried out with an initial amount of free gas which also accounts gas release during gaseous cavitation. The current study used continuity equation in gaseous phase, mixture continuity equation and mixture momentum equation.

The continuity equation for gaseous phase (1), mixture continuity equation (2) and mixture momentum equation (3) for homogeneous air- water mixture are given by [2],

$$\frac{\partial m}{\partial t} = \frac{\beta}{\theta_m RT}[p_s - p] \tag{1}$$

$$\frac{\partial \phi}{\partial t} + \frac{c^2}{gA}\frac{\partial Q}{\partial x} = 0 \tag{2}$$

$$\frac{\partial u}{\partial t} + \frac{\partial H}{\partial x} + \frac{1}{r}\frac{\partial (r\tau_w)}{\partial r} = 0 \tag{3}$$

In which, β is the Henry's law constant, θ_m is the relaxation time for gas release, R is the universal gas constant, T is the absolute temperature and p_s is the saturation pressure. ϕ is an auxiliary variable defined as given below

$$\phi = \frac{p}{\rho g} - \frac{c^2 mRT}{pg} + \frac{mc^2}{\rho g} \tag{4}$$

Here, ρ and c represents the mixture density and mixture wave velocity (because for gaseous cavitation modelling, the liquid is considered as a mixture of gas and liquid). Then the expressions for these quantities are given below.

$$\rho = [1 - \frac{mRT}{p}]\rho_w + \frac{mRT}{p}\rho_g \tag{5}$$

Where, ρ_w is the density of water, ρ_g is the density of gas, m is the mass of free gas per unit volume, R is the universal gas constant, T is the absolute temperature.

$$c = \sqrt{\frac{K_m}{\rho}} \tag{6}$$

Where, K_m is the mixture bulk modulus and it is given by the equation,

$$K_m = \frac{1}{\frac{1}{K_l} + \frac{\alpha_0 p_0}{p^2} + \frac{D}{E_C e}} \tag{7}$$

Where, K_l is the bulk modulus of water, α_0 is the initial volume fraction of gas, p_0 is the initial pressure, E_c is the modulus of elasticity of conduit, D is the diameter and e is the pipe thickness.

For turbulent flow, the shear stress in (3) can be calculated by a two-zone turbulence model[6]. According to this model, Newton's law is used in the viscous sublayer and mixing length model is used in the turbulent core. For this model, the thickness of viscous sublayer is to be calculated and it is obtained as the distance from the wall to the intersection between the velocity profiles in the viscous sublayer and in the turbulent zone (assuming that velocity profile is linear in the viscous sublayer and logarithmic in the turbulent region)[7].

3. Numerical Computation

The pipe is divided into cylindrical grid elements having a fixed length Δx in longitudinal direction and constant area ΔA in radial direction. Velocities are calculated at the centre of each radial mesh, and shear stress is calculated for the internal and external sides of the radial mesh (Fig. 1). Variables such as pressure head H, mass m, and ϕ etc. are defined at each grid point and vary along longitudinal direction.

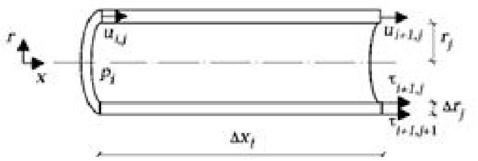

Fig. 1. Cylindrical grid element [2]

The numerical scheme adopted for the solution of the governing equations is the MacCormack approach, which is a finite difference method introduced by R.W. MacCormack in 1969. This method consists of two steps; a *predictor* step which is followed by a *corrector* step.

3.1. Predictor Step[2]

$$\frac{\phi_i^{\ p}-\phi_i^{\ k}}{\Delta t}+\frac{c^2}{gA}\frac{Q_{i+1}^{\ \ k}-Q_i^{\ k}}{\Delta x} \tag{8}$$

$$\frac{m_i^{\ p}-m_i^{\ k}}{\Delta t}=\frac{\beta}{\theta_m RT}(p_{si}^{\ \ k}-p_i^{\ k}) \tag{9}$$

$$\frac{u_{i,j}^{\ \ p}-u_{i,j}^{\ \ k}}{\Delta t}+g\frac{H_{i+1}^{\ \ k}-H_i^{\ k}}{\Delta x}+\frac{2\pi}{\rho}\frac{(r_{j+1}\tau_{wi,j+1}-r_j\tau_{wi,j})}{\Delta A}=0 \tag{10}$$

3.2. Corrector Step [2]

$$\frac{\phi_i^{\ c}-\phi_i^{\ k}}{\Delta t}+\frac{c^2}{gA}\frac{Q_i^{\ p}-Q_{i-1}^{\ \ p}}{\Delta x} \tag{11}$$

$$\frac{m_i^{\ c}-m_i^{\ k}}{\Delta t}=\frac{\beta}{\theta_m RT}(p_{si}^{\ \ p}-p_i^{\ p}) \tag{12}$$

$$\frac{u_{i,j}^{\ \ c}-u_{i,j}^{\ \ k}}{\Delta t}+g\frac{H_i^{\ p}-H_{i-1}^{\ \ p}}{\Delta x}+\frac{2\pi}{\rho}\frac{(r_{j+1}\tau_{wi,j+1}-r_j\tau_{wi,j})}{\Delta A}=0 \tag{13}$$

The *x* direction is denoted by the subscript *i*, the y direction by the subscript *j*, and the time domain by the superscript *k*. The time level where the flow variables are known is denoted by superscript *k*, and the unknown time level is represented by the notation *k+1*. In the momentum equation $\tau_{i,j}$ and $\tau_{i,j+1}$ are the internal and external shear stresses respectively. The value at time *k+1* is computed as the average of initial and corrector values of the variables.

4. Experimental Setup

The experimental installation consists of a zinc-plated steel pipe (internal diameter 53.9 mm, thickness 3.2 mm, modulus of elasticity 2.06x10 ^{11}N/m^2, roughness 0.1 mm, length 144 m) which is fed by a centrifugal pump [2]. At the downstream end of the pipe, a pressure tank is located and at the upstream end there is a ball valve. The line pressure is measured by strain gauge pressure transducers, having a range of 0–10 bar. Discharge is measured by an electromagnetic flowmeter with adjustable full scale velocity, with maximum errors of ±0.1% of full scale.

The experiment is done by closing the valve at the upstream end of the pipe in 0.04s manually and the simulations is carried out for 5s. The initial discharge and free gas amount are 0.00068 and 33Mg/m^3 respectively. Table. 1 shows the calibrated parameters used for simulation [2].

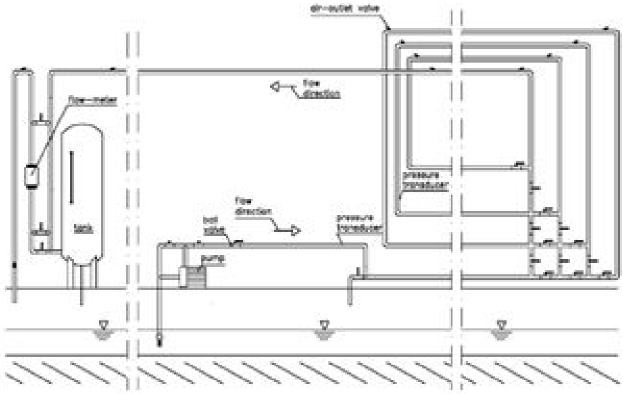

Fig. 2. Experimental setup [2]

Table 1. Calibrated parameters used [2]

β	θ_m (s)
0.02	440

5. Results and Discussions

Results are described for two cases, one for constant wave velocity and other for variable wave velocity.

5.1. Case I (Modelling with constant wave velocity)

In this model, the continuity and momentum equations for the liquid gas mixture is taken along with the gas continuity equation. An initial amount of free gas, 33 mg/m^3 is considered [2]. Since there is an initial amount of free gas, the variation in the density, bulk modulus, wave velocity etc. are taken in the analysis. The computed pressure head and gas release at the valve are shown below (Fig. 3 and Fig. 4)

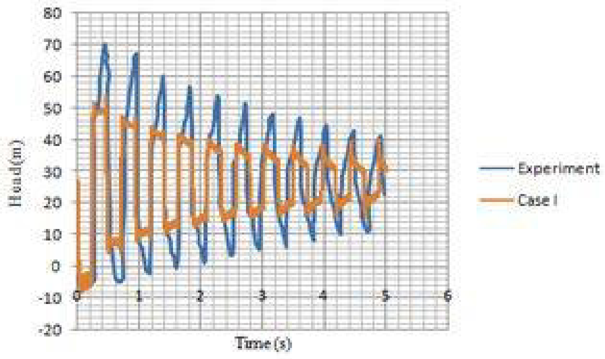

Fig. 3. Comparison of pressure head at valve

When initial gas mass is taken for modelling, computed pressure wave shows more damping with respect to the experimental pressure heads. Though, the initial pressure head drop is coinciding well with the experimental result, following pressure heads show significant reduction in magnitude, both in crest and troughs.

Since the flowing liquid is a mixture of liquid and gas (both free and released gas) this mismatching in observation may be due to the lack of representation of variable wave velocity.

When gas mass curve is plotted (Fig. 4) for the same case, it is found that they are nearly matching only in the first peak. As the time progresses the trend of mass curve is converging well, but the values are not coinciding. These uncertainties may be also due to the assumption of constant wave velocity, without considering the change in velocity due to instantaneous change in the gas release during gaseous cavitation. Therefore for the next case, variable wave velocity of pressure wave is considered.

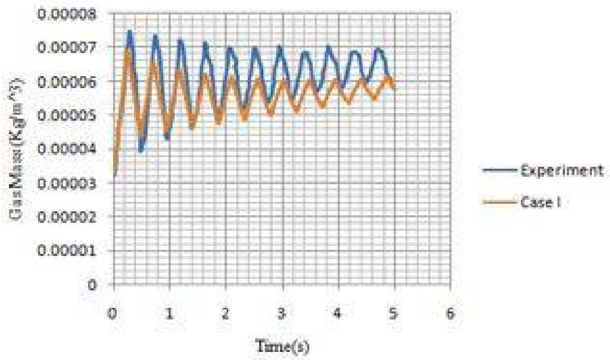

Fig. 4. Comparison of gas mass curve

5.2. Case II (Modelling with variable wave speed)

In this model, the mixture continuity equation, mixture momentum equation and gas continuity equation are used with an initial amount of free gas ($33 mg/m^3$). The variable wave velocity is considered for this case. The computed variable wave speed accounting the variation in density and bulk modulus due to the presence of gas, is plotted and is shown in Fig. 5.

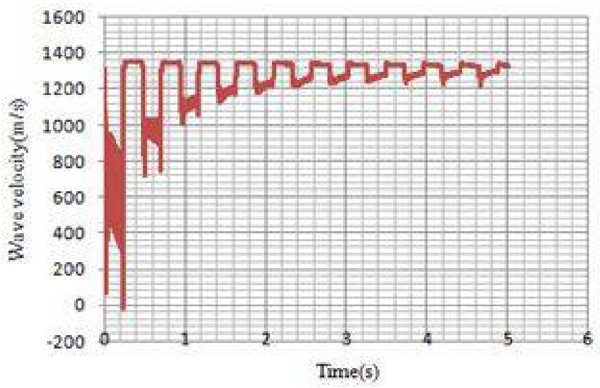

Fig. 5. Variable wave speed

Analyzing the curve of wave velocity (Fig. 5), one can see a drastic change in wave velocity in a very short time period.

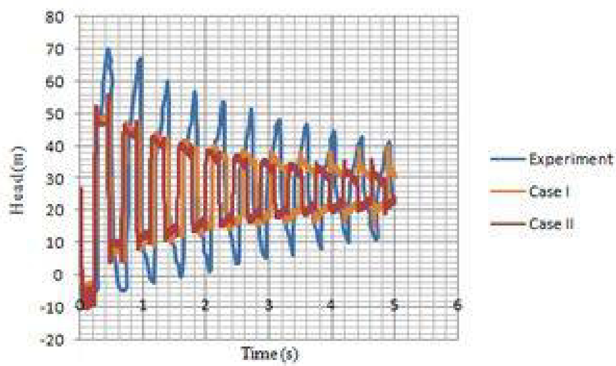

Fig. 6. Comparison of pressure head at valve

Fig. 6 shows the comparison of pressure heads at the valve for the models I and II and for the experiment. Though the pressure drop at the beginning match well in both the models, pressure hikes are found to be less than the actual experimental peaks in both cases. Higher magnitude is observed for the Case II model with respect to case I model. This may be due to the incorporation of variable wave velocity. Case I and Case II model shows more attenuation than that of the experimental pressure heads. But when the gas mass curve is plotted (Fig. 7), results from Case II model (with variable wave speed) gives better result.

Even if the model can explain the water hammer pressure drop, extra damping is observed in the subsequent peaks, when compared with the experimental pressure heads.

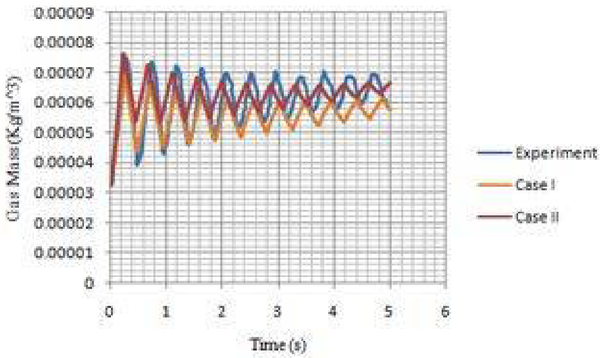

Fig. 7. Comparison of gas mass at valve

From the comparison of gas mass curves (Fig. 7), it is found that the model which considers initial gas mass, gasesous cavitation, and variable wave velocity, shows good agreement with the experimental results. When combining the observations of pressure head and gas release, Case II model can be used to represent gaseous cavitation.

6. Conclusion

The current study investigates the effect of energy dissipation and variable wave speed in the modelling of transient gasesous cavitation. It is found that the model which accounts for variable wave speed is able to predict gas release and pressure variation in a better way. However, further improvement in the model is essential to get rid of the spurious damping effect in the pressure oscillations.

References

[1] Taieb EH, Lili T. Validation of Hyperbolic Model for Water-Hammer in Deformable Pipes. J Fluids Engineering (ASME) 2000: 122: 57-64.

[2] Cannizaro D, Pezzinga G. Energy dissipation in transient gaseous cavitation. J Hydraulic Engineering 2005: 131: 724-732.

[3] Sumam KS, Thampi GS, Sajikumar N. An Alternate Approach for Modelling of Transient Vaporous Cavitation. International Journal for Numerical Methods in Fluids. [Online], AvailableHTTP: http://www.interscience.wiley.com. 2009

[4] Adamkowski A, Lewandowski M. Investigation of Hydraulic Transients in a Pipeline with Column Seperation. J Hydraulic Engineering 2012: 138: 935-944.

[5] Cannizaro D, Pezzinga G. Analysis of Transient Vaporous Cavitation in Pipes by a Distributed 2D model. J Hydraulic Engineering 2014: 140: 1-9.

[6] Cannizaro D, Pezzinga G. Two Dimensional Features of Viscoelastic Models of Pipe Transients. J Hydraulic Engineering 2014.

[7] Modi PN, Seth SM. Hydraulics and Fluid Mechanics Including Hydraulic Machines. Standard Book House. 2005